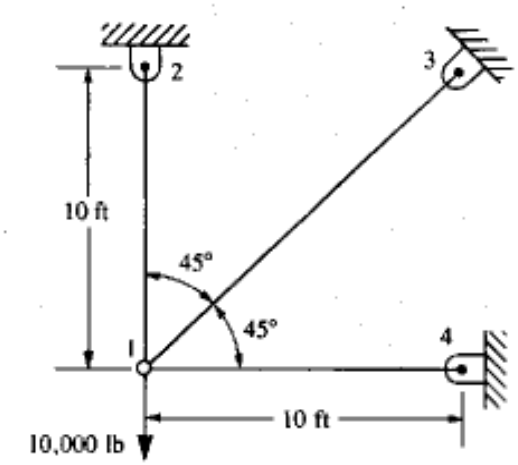
# ABAQUS Tutorial – 2D Truss Analysis

Problem: Determine the nodal displacements and element stresses for the truss shown below (ref. Logan's text example 3.5 pp 81-84). Use E=30x10$^6$ psi and A=2 in$^2$. Compare to text solution: $d_{1x}$ = 0.414e-2 in , $d_{1y}$ = -1.59e-2 in, element stresses = -1035, 1471, and 3965 psi.




Start => All Programs => Abaqus 6.8-1 => Abaqus CAE => Create Model Database
File => Set Working Directory => Browse to find desired S: drive directory => OK

Module: Sketch

Sketch => Create => Name: truss-demo => Continue
Add=> Point => enter coordinates (0,0), (120,0), (120,120), (0,120) => select 'red X'
View => Auto-Fit
Add => Line => Connected Line => select (0,0) node with mouse, then (120,0) node, right click => Cancel Procedure
Add => Line => Connected Line => select (0,0) node with mouse, then (120,120) node, right click => Cancel Procedure
Add => Line => Connected Line => select (0,0) node with mouse, then (0,120) node, right click => Cancel Procedure=> Done

Module: Part

Part => Create => select 2D Planar, Deformable, Wire => Continue
Add => Sketch => select 'truss_demo' => Done => Done

Module: Property

Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 30e6, Poisson's ratio = 0.3 => OK
Section => Create => Name: Section-1, Beam, Truss => Continue => set Material: Material-1, Cross-sectional area: 2

Assign Section => select all elements by dragging mouse => Done => Section-1 => OK => Done

Module: Assembly

Instance => Create => Part-1 => Independent (mesh on instance) => OK

Module: Step

Step => Create => Name: Step-1, Initial, Static, General => Continue => accept default settings => OK

Module: Load

Load => Create => Name: Step-1, Step: Step 1, Mechanical, Concentrated Force => Continue => select node at (0,0) => Done => set CF2: -10000 => OK
BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select nodes at (120,0), (120,120) and (0,120) using SHIFT key to select multiple nodes => Done => set U1: 0 and U2: 0

Module: Mesh

Seed => Edge by Number => select entire truss by dragging mouse => Done => Number of elements along edges: 1 => press Enter => Done
Mesh => Element Type => select entire truss by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear: Family: Truss => OK => Done
Mesh => Instance => OK to mesh the part Instance: Yes

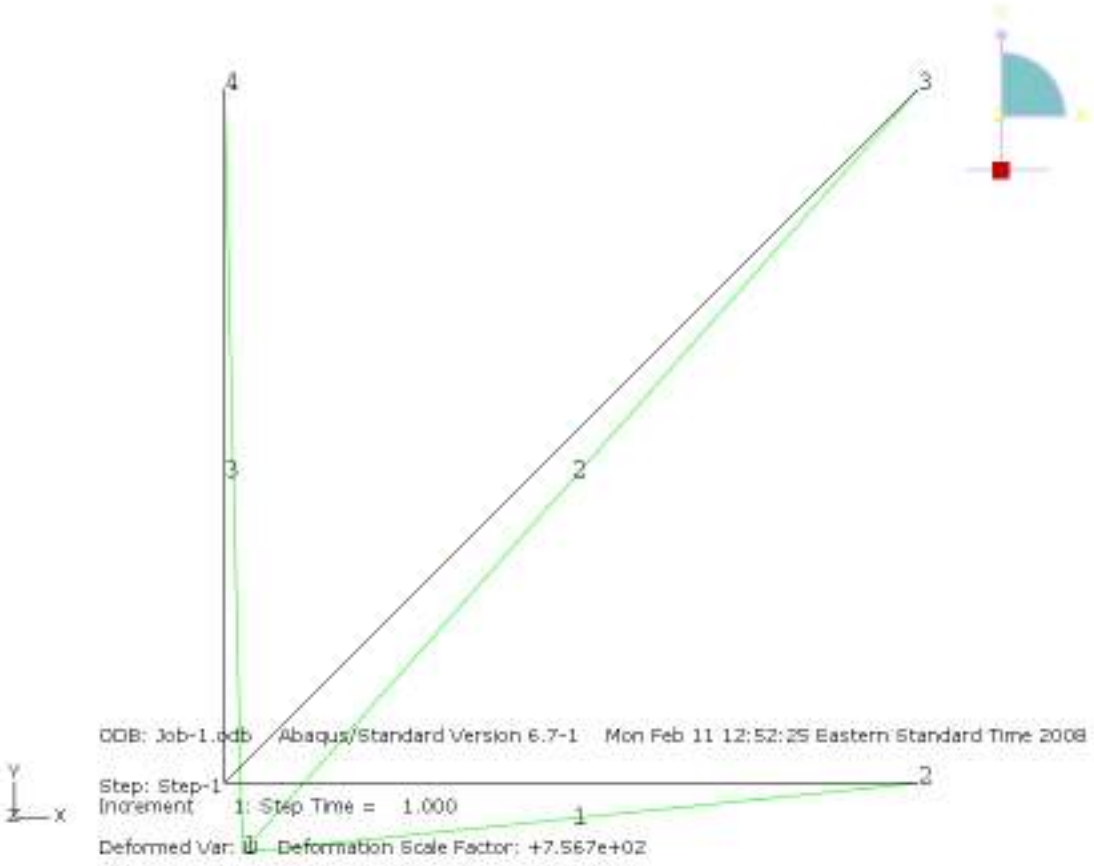
Module: Job

Job => Create => Name: Job-1, Model: Model-1 => Continue => Job Type: Full analysis, Run Mode: Background, Submit Time: Immediately => OK
Job => Submit => Job-1
Job => Manager => Results (enters Module: Visualization)
Plot = > Allow multiple plot states
Plot => Undeformed Shape
Plot => Deformed Shape
Plot => Contours => On Deformed Shape
Result => Options => unselect "Average element output at nodes"
View => Graphics Options => Viewport Background = Solid=> Color => White (click on black tile to change background color)
Options => Common => Labels => select 'Show element labels' and 'Show node labels', set label colors to black => OK
Ctrl-C => Copies graphics window to clipboard => Paste in MS Word, etc.
Report => Field Output => Variable => Position: Unique Nodal => select U: Spatial Displacements => Apply => Unselect U
Report => Field Output => Variable => Position: Centroid => select S: Stress Components => Click on '>' and unselect all stresses except S11 => Apply

Open file ‘abaqus.rpt’ and cut and paste desired results into MS Word
File => Save => enter desired file name (Abaqus will append .cae)
File => Exit

Results:

Deformed Mesh:




Tabulated Results (using cut and paste from abaqus.rpt)

Nodal displacements:

| Node Label | U.Magnitude @Loc 1 | U.U1 @Loc 1 | U.U2 @Loc 1 |
|---|---|---|---|
| 1 | 16.3899E-03 | 4.14214E-03 | -15.8579E-03 |
| 2 | 2.07107E-33 | 2.07107E-33 | 0. |
| 3 | 2.92893E-33 | -2.07107E-33 | -2.07107E-33 |
| 4 | 7.92893E-33 | 0. | -7.92893E-33 |
| Minimum | 2.07107E-33 | -2.07107E-33 | -15.8579E-03 |
| At Node | 2 | 3 | 1 |
| Maximum | 16.3899E-03 | 4.14214E-03 | 0. |

| | | | |
|---|---|---|---|
| At Node | 1 | 1 | 2 |
| Total | 16.3899E-03 | 4.14214E-03 | -15.8579E-03 |

Element Stresses:

| Element Label | S.S11 @Loc 1 |
|---|---|
| 1 | -1.03553E+03 |
| 2 | 1.46447E+03 |
| 3 | 3.96447E+03 |
| Minimum | -1.03553E+03 |
| At Element | 1 |
| Maximum | 3.96447E+03 |
| At Element | 3 |
| Total | 4.39340E+03 |